Developing DSP Code

Author: Donald Eubanks

Hanzi Yang

Trinh Dinh

Jeff Brower

Copyright © Signalogic, Inc.

July 2002

Revision 1.1

Table of Contents

1.0 Installation of Texas Instruments Code Composer Studio v2.1 (CCS) 2
1.1. Cleaning the System 2
1.2. Creating a Merged Installation of Code Composer Studio v2.1 3
1.3. Steps to Install 3
2.0 Installation of SigC54xx, SigC55xx or SigC67xx Hardware 5
3.0 Installation of Spectrum Digital XDS510PP JTAG Emulator 6
3.1 Installation of Spectrum Digital XDS510PP JTAG Emulator Drivers for CCS 6
3.1.1 Download 6
3.1.2 Installation 6
3.2 Installation of Spectrum Digital XDS510PP JTAG Emulator Software 6
3.2.1 Installation 6
3.2.2 Run 7
3.2.3 Test 7
4.0 Configuring Code Composer Studio to Run with the Emulator 8
4.1 Run 8
4.2 Verify 8
4.3 Run DirectDSP or Hypersignal 8
4.4 Run Code Composer Studio 8
4.5 Debugging 9
5.0 Signalogic C54xx and C6xxx Source Code Interface Development 10
5.1 Host Mode and UserProc Debugging 14
5.2 Stand-Alone Mode and UserProc Debugging 17
6.0 Code Composer Studio Debugging Examples 18
7.0 DirectDSP Development Examples 19
5.1 Visual Studio Development Examples 19
5.2 MATLAB Development Examples 22
8.0 Hypersignal Software Example 24

1.0 Installation of Texas Instruments Code Composer Studio v2.1 (CCS)

1.1. Cleaning the System

Clean the host system of any existing TI tools, or any CCS for any of the DSP series. Here are the steps to clean the CCS from the system.

(BEFORE PROCEEDING PLEASE NOTE: Errors in the registry can prevent your PC from booting properly. You should make a backup of your registry just in case something is done incorrectly that causes problems with your system. If you are not comfortable modifying the registry, contact your computer support group for assistance.) This information is provided as a service from Texas Instruments. However, Texas Instruments assumes NO liability for problems caused by incorrectly modifying the registry.

Please follow these steps to completely remove CCS from your machine.

- Go through the Add/Remove programs step in Control panel to remove Code Composer Studio.
- When the un-installation is complete, check to see if there is a "details" button to the left of the "OK" button. If a "details" button does exist, please click on it and look at the directories that are remaining and then delete them from your drive.
- Next, go to the Start menu and select Run. Type “regedit” and click OK. (This will cause the Registry Editor window to come up.) Next, go to the following places in the registry and delete the Go DSP, TI and Texas Instruments keys (if they continue to exist).

  HKEY_CURRENT_USER -> Software -> Go DSP

  HKEY_CURRENT_USER -> Software -> Texas Instruments

  HKEY_CURRENT_USER -> Software -> TI

- In the registry now go to HKEY_LOCAL_MACHINE -> SOFTWARE. From here delete the GO DSP and TI and Texas Instruments keys (if they exist).

  HKEY_LOCAL_MACHINE -> Software -> Go DSP

  HKEY_LOCAL_MACHINE -> Software -> Texas Instruments

  HKEY_LOCAL_MACHINE -> Software -> TI

After following these steps, reboot your machine. By eliminating all of the references to Code Composer Studio from your computer, you will be installing Code Composer Studio on a clean machine.

- Must be the full version with all needed drivers. Please note that the CCS version offered on TI's web site (30 day trial version) will not work since it does not include all necessary drivers needed to work with the Spectrum Digital Emulator. Also note that a "DSK" version of CCS (a copy of CCS supplied with a DSP Starter Kit) will NOT work with the Spectrum Digital Emulator.

NOTE: If you have any Signalogic programs installed on your system, you MUST take the following steps to ensure they continue to run correctly after the CCS installation.

a. Click the Start button.

b. Choose "Run".

c. Type "sysedit" in the pop-up window

d. The System Configuration Editor program will load, with several windows already open inside.

e. Click the "system.ini" window.

f. Scroll through the contents of the system.ini file until you reach the section of the file titled "[386enh]".

g. Below the "[386enh]" title, add the line "device = pipe.vxd"or ensure this line is already there. (This line was deleted during the CCS installation)

h. Save the system.ini file, and close the System Configuration Editor.

NOTE: If you are installing only one version of CCS v2.1 (either the C5000 or C6000 version), or are installing a CCS other than version v2.1, skip to the Installation of Spectrum Digital XDS51DPP JTAG Emulator Hardware section below.

1.2. Creating a Merged Installation of Code Composer Studio v2.1

It is possible to install both the C5000 and C6000 versions of CCS v2.1 to create a "C5000/C6000" installation of CCS v2.1. This unified version allows development for all C5000 and C6000 series processors on one machine. The instructions of creating a unified installation follow:

1.3. Steps to Install

Insert the CD for the other processor series version of Code Composer Studio. Choose "Install Code Composer Studio" from the menu. After you accept the licensing agreement, the program will offer two options. Choose the "Merge, Upgrade, or Reinstall

an Existing Installation” option. If the CD in the drive is a separate version of CCS v2.1 than the one already installed, this option will turn the previous installation into a combined installation.

2.0 Installation of SigC54xx, SigC55xx or SigC67xx Hardware

For installation of SigC54xx, SigC55xx, and SigC67xx hardware, please view the following FTP site link:

ftp://ftp.signalogic.com/VDS/VDS Documentation.pdf

For additional information about installation of SigC54xx, please click on this link:

http://www.signalogic.com/tech_support/melp_support.htm#Speech

For installation of SigC67xx, please click on these links:

http://www.signalogic.com/tech_support/m67_support.htm#m62install

http://www.signalogic.com/tech_support/m67_support.htm#m67install

http://www.signalogic.com/tech_support/m67_support.htm#m67issues

3.0 Installation of Spectrum Digital XDS510PP JTAG Emulator

- Connect the emulator to a wall outlet with the provided cabling.
- Connect the emulator to the parallel port on the PC using the provided parallel port cables.
- Connect the emulator's JTAG connector to the target board's JTAG header, and power-up the board.

***NOTE**: If the board is connected to a PCI or ISA slot, power-down the computer before connecting the emulator.*

3.1 Installation of Spectrum Digital XDS510PP JTAG Emulator Drivers for CCS

3.1.1 Download

Download the XDS510PP drivers for use with Code Composer Studio. The drivers can be found at:

http://www.spectrumdigital.com/ftp.asp

***NOTE**: There are different drivers for different (a) versions of CCS and (b) processor families, be sure to choose the correct drivers for your system.*

3.1.2 Installation

Run the downloaded .exe file and follow all installation steps. For added simplicity, be sure to use the default directory (C:\TI) for the drivers (assuming that CCS was installed to C:\TI as well).

3.2 Installation of Spectrum Digital XDS510PP JTAG Emulator Software

3.2.1 Installation

Install the Spectrum Digital Configuration utility (SDConfig). Like the drivers, SDConfig can be downloaded from:

http://www.spectrumdigital.com/ftp.asp

3.2.2 Run

Run SDConfig after installation. Choose the icon representing the port to which the emulator is connected (LPT1, the PC's main printer port is located at 0x378) and in the right-hand window, double-click "Emu." Select your Spectrum Digital product from the drop-down menu (be sure to pay careful attention to whether or not you have an XDS510PP or an XDS510PP Plus). Choose the emulator port mode (SPP8 should work for all possible port modes. If SPP8 does not work, check this document for information on determining your port address and mode using Spectrum Digital's port I/O utility:

http://www.spectrumdigital.com/technical/pdfs/setup_chek.pdf

3.2.3 Test

Once the configuration is complete, reset the emulator (press the button with the big red "R" in the configuration utility). You should see a message at the bottom of the program "**Emulator is reset." If you do not see this message, verify your hardware connections and try again. Once the emulator has been reset, run the emulator test by clicking the button with an "emu" in front of a check mark. If the test passes, CCS setup can proceed. If either the reset or emulator test failed, reference this document for information and answers to frequently-asked questions:

http://www.spectrumdigital.com/technical/pdfs/faq.pdf

4.0 Configuring Code Composer Studio to Run with the Emulator

4.1 Run

Run the setup program installed with CCS. Clear any existing configuration once you enter Setup (There should be no green board icons in the left-most frame. From the right-most frame, select "Install a Device Driver." The file selection box will open to the c:\zzzzz\drivers directory (where zzzzz is the directory to which CCS was installed) Select the file named "sdgoXXXX.dvr" (where XXXX is representative of the processor family to which the target board's processor belongs.) CCS Setup will then ask you to name this device configuration. After the configuration is named, new board icon will appear in the middle frame (with whatever name you assigned it) Select this configuration. CCS Setup will again ask you to name the configuration. In addition, it will ask for the I/O port it is on. **The I/O port entered here MUST match the port to which the emulator is connected!**

4.2 Verify

After the I/O port is entered, add processors to the configuration by highlighting them and selecting "Add Single" or "Add Multiple." (Obviously it is important to choose the processor types that correspond to the processors on the board.)

NOTE: If there are multiple processors on the board of different types, the processors must be added to the configuration in the order they are laid out on the JTAG scan chain.

If Signalogic provided a GEL file with the Source Code Interface, make sure to add it to the "Startup GEL File(s)" tab. After all options are entered, the board configuration and processors added will be in the left-most frame in CCS Setup. From the File menu, choose "Save." CCS is now ready to use with the Spectrum Digital Emulator.

4.3 Run DirectDSP or Hypersignal

Run any Signalogic DirectDSP® or Hypersignal® program to initialize the DSP and prepare it for JTAG use. (see sections 7.0 – 9.0, for Visual Studio, MATLAB, and Hypersignal software examples).

4.4 Run Code Composer Studio

Run Code Composer Studio. There should be no errors at start-up. If a dialog box appears with the error message "Cannot initialize target DSP," then verify that the driver selected is the correct one for the processor type, and that the Port I/O field for the configuration matches that of SDConfig.

4.5 Debugging

Once CCS is running, all JTAG debugging functionality will be enabled for the board entered in step 1. For more advanced SCI development options and common CCs debugging functions, see section 6.0.

5.0 Signalogic C54xx and C6xxx Source Code Interface Development

This section covers installation and use of Signalogic's "DSP Source Code Interface" software, or SCI software, SCI versions supported under TI Code Composer Stude (CCS) include C54xx, C55xx, C62xx, and C67xx. SCI versions supported only by DSP vendor command-line tools include C3x, C4x, 563xx, and ADSP-2106x.

1. Install Code Composer Studio v2.0 or v2.1 for C5000 and/or C6000 series. Only CCS v2.0 or higher is supported for SCI development.

2. Click on CCS 2 icon on the desktop or go through the Start menu.

3. If the following message box (Figure Figure 5-1) appears, click the Ignore button.

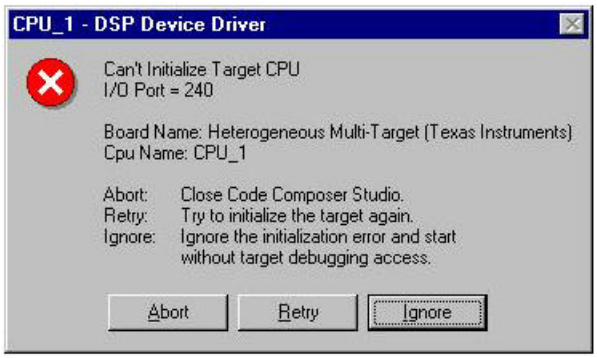
CPU_1 - DSP Device Driver
Can't Initialize Target CPU
I/O Port = 240
Board Name: Heterogeneous Multi-Target (Texas Instruments)
Cpu Name: CPU_1
Abort: Close Code Composer Studio.
Retry: Try to initialize the target again.
Ignore: Ignore the initialization error and start without target debugging access.
Abort
Retry
Ignore

Figure 5-1

4. To open the C54xx SCI project,
    a. Project -> Open
    b. Browse to \dspcode\c54xx\Sci directory.
    c. Open C54xx_SCI.pjt
5. To open the C6x11 SCI project,
    a. Project -> Open
    b. Browse to \dspcode\c6xxx\Dsk_C6x11 directory.
    c. Open tmsc6xxx.pjt

6. To open the M62_M67 SCI project,
    a. Project -> Open
    b. Browse to \dspcode\c6xxx\M62_M67 directory.
    c. Open tms6xiim.pjt
7. To build the project, select Project -> build or click on the build icon on the toolbar.
8. The resulting .out file (COEFF, or executable file) out file should be located at,
    a. C54xx SCI → \dspcode\c54xx\C54xx_SCI\tmsc54xx.out.
    b. C6x11 SCI → \dspcode\c6xxx\Dsk_C6x11\tmsc6x11.out.
    c. M62_M67 SCI → \dspcode\c6xxx\M62_M67\tms6XIIm.out
9. To open the Source Code Interface in Code Composer Studio v2.1, select “Open” from the Project menu. (See Figure 5-2)

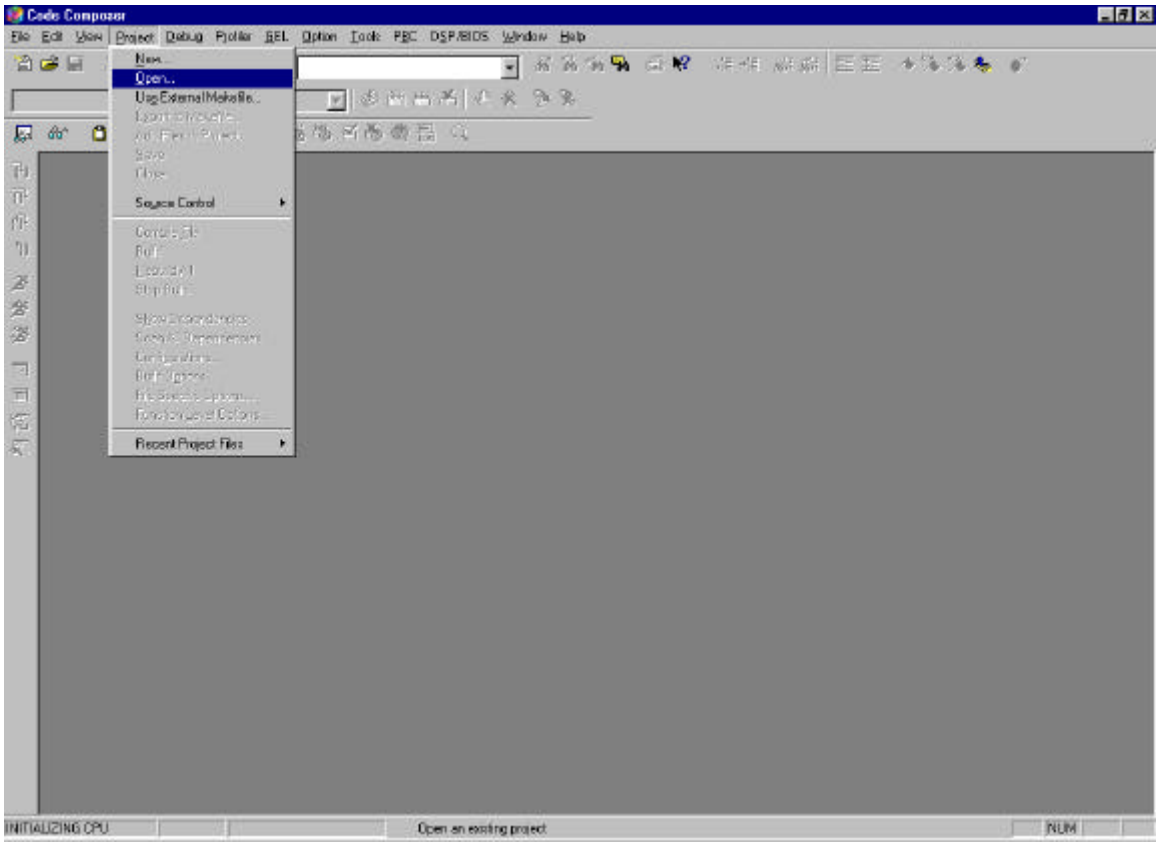

Figure 5-2 Opening a project in CCS

10. Double-click the supplied CCS v2.1 project file (**.pjt**). The project view window will display the name of the project and all files associated with the project using the standard Windows expanding tree display (clicking the '+' will display the contents of a project or directory, clicking the '-' will collapse the contents into the project or directory). (See Figure 5-3)

Note: If you are running a version of Code Composer Studio lower than v2.0, the steps for opening a project still apply, however the extension for older project files is .mak not .pjt.

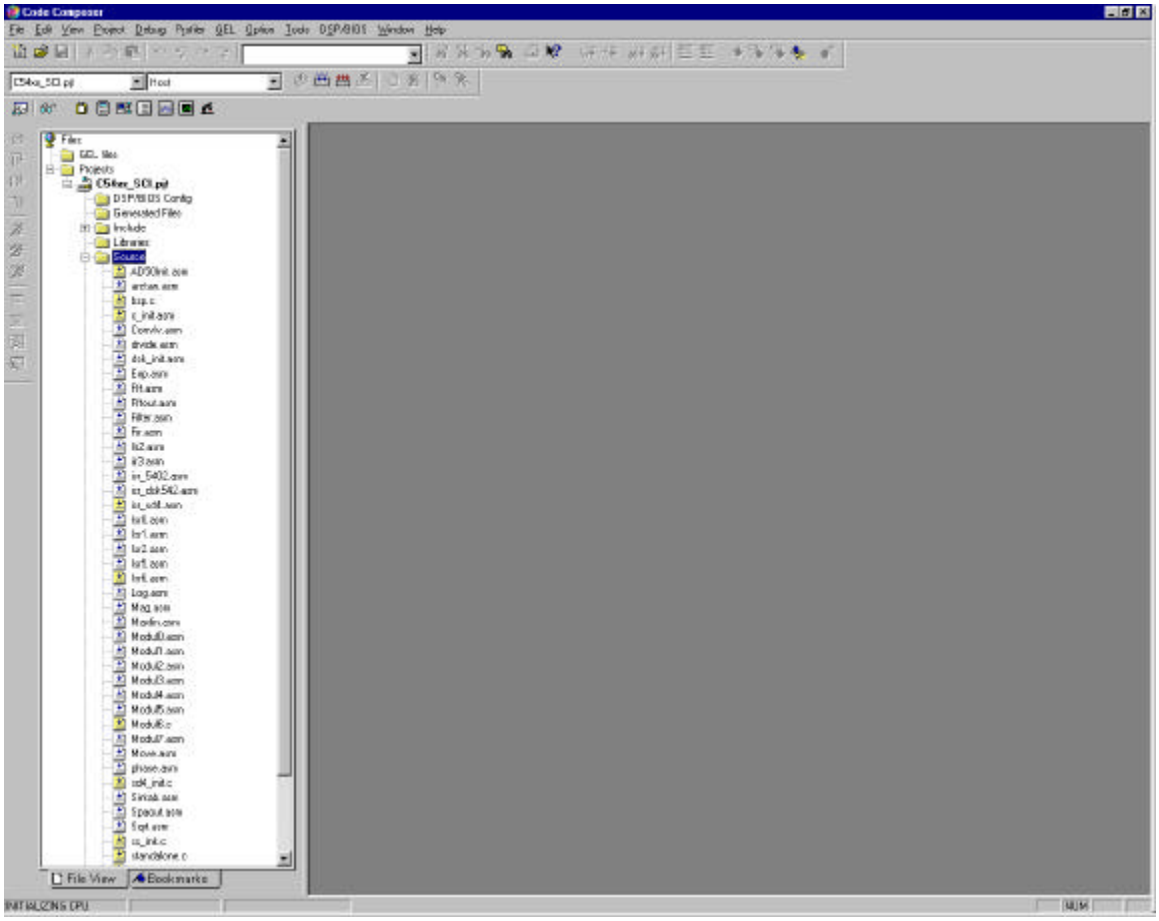

Figure 5-3 SCI project loaded

11. Double-clicking files in the project display will open them in the File Display window. (See Figure 5-4 below)

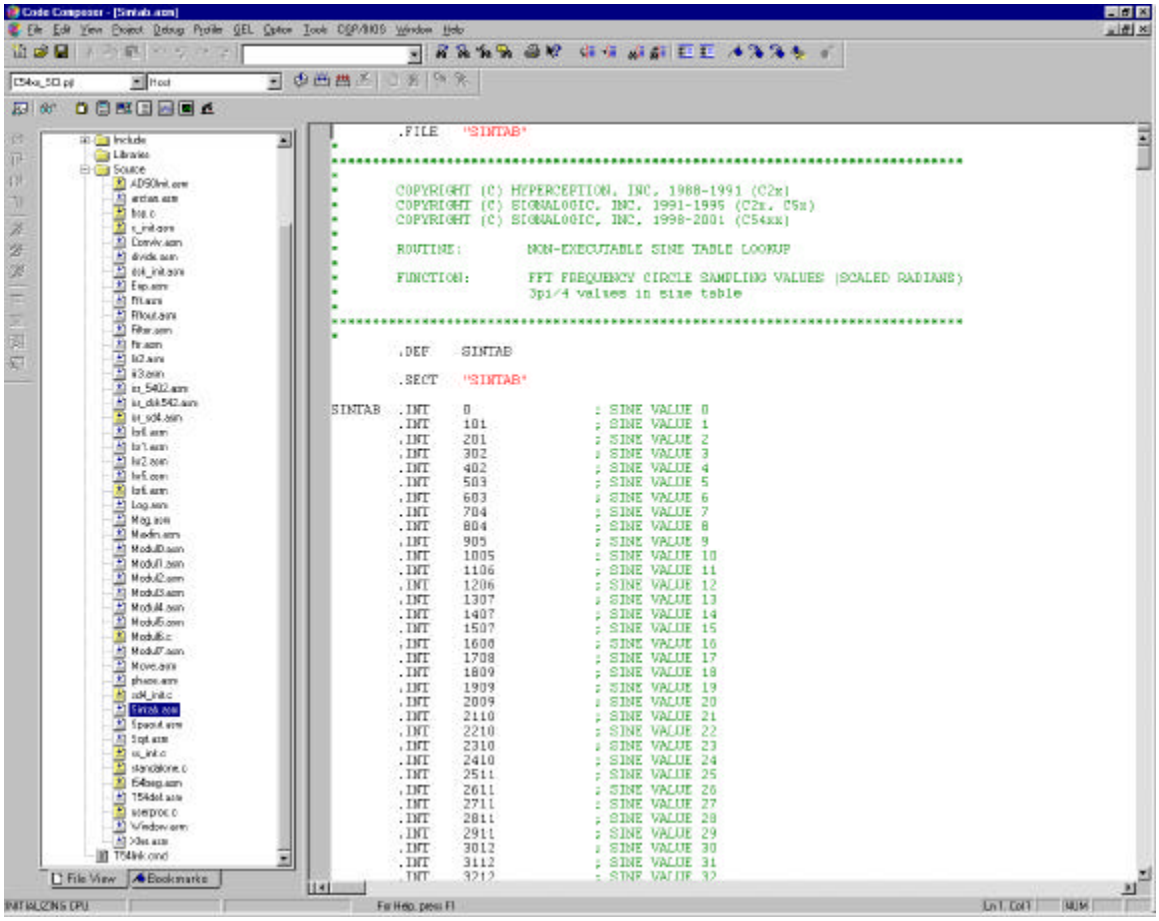
.FILE "SINTAB"
COPYRIGHT (C) HYPERCEPTION, INC. 1988-1991 (C2x)
COPYRIGHT (C) SIGNALOGIC, INC. 1991-1995 (C2x, C5x)
COPYRIGHT (C) SIGNALOGIC, INC. 1998-2001 (C54xx)
ROUTINE: NON-EXECUTABLE SINE TABLE LOOKUP
FUNCTION: FFT FREQUENCY CIRCLE SAMPLING VALUES (SCALED RADIANS)
3pi/4 values in sine table
.DEF SINTAB
.SECT "SINTAB"
SINTAB .INT 0 ; SINE VALUE 0
.INT 101 ; SINE VALUE 1
.INT 201 ; SINE VALUE 2
.INT 302 ; SINE VALUE 3
.INT 402 ; SINE VALUE 4
.INT 503 ; SINE VALUE 5
.INT 603 ; SINE VALUE 6
.INT 704 ; SINE VALUE 7
.INT 804 ; SINE VALUE 8
.INT 905 ; SINE VALUE 9
.INT 1005 ; SINE VALUE 10
.INT 1106 ; SINE VALUE 11
.INT 1206 ; SINE VALUE 12
.INT 1307 ; SINE VALUE 13
.INT 1407 ; SINE VALUE 14
.INT 1507 ; SINE VALUE 15
.INT 1608 ; SINE VALUE 16
.INT 1708 ; SINE VALUE 17
.INT 1809 ; SINE VALUE 18
.INT 1909 ; SINE VALUE 19
.INT 2009 ; SINE VALUE 20
.INT 2110 ; SINE VALUE 21
.INT 2210 ; SINE VALUE 22
.INT 2310 ; SINE VALUE 23
.INT 2410 ; SINE VALUE 24
.INT 2511 ; SINE VALUE 25
.INT 2611 ; SINE VALUE 26
.INT 2711 ; SINE VALUE 27
.INT 2811 ; SINE VALUE 28
.INT 2911 ; SINE VALUE 29
.INT 3012 ; SINE VALUE 30
.INT 3112 ; SINE VALUE 31

Figure 5-4 File opened

For both the C54xx and C6xxx Source Code Interfaces, Signalogic has implemented "Host" and "Stand-Alone" modes to assist users with algorithm and UserProc function debugging under:

- Host based development environment, using Direct DSP software
- JTAG-only development environment

DirectDSP software supports Visual Studio, MATLAB and LabVIEW software, and provides an integrated development and delay environment for these popular programming methods. For more information on DirectDSP software, see section 7.0.

JTAG is most often used with CCS software and provides an extensive low-level debug capability, suitable for hardware development and low-level software development.

5.1 Host Mode and UserProc Debugging

1. Project → Configurations. You will see the following dialog box as in Figure 5-5 below.

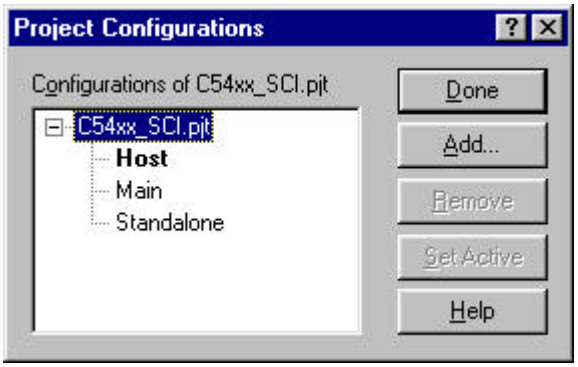

Figure 5-5 CCS Project Configuration Dialog Box

2. Select Host and click Set Active button.
3. Click Done.
4. Project → Build or F7 to build the project.
5. The .out, .map, and .obj files will be located in the same directory as the .pjt files.
6. Copy the .out file to \hsmacro directory.
7. The new .out file can be tested with DSPower-Debugger
8. Start → Programs → Signalogic DSP Software → DSPower-Debugger.
9. You will see the Hardware Manager dialog box as in Figure 5-6 below.
10. Choose the proper board type and click Ok, you will see the Digital Scope program.

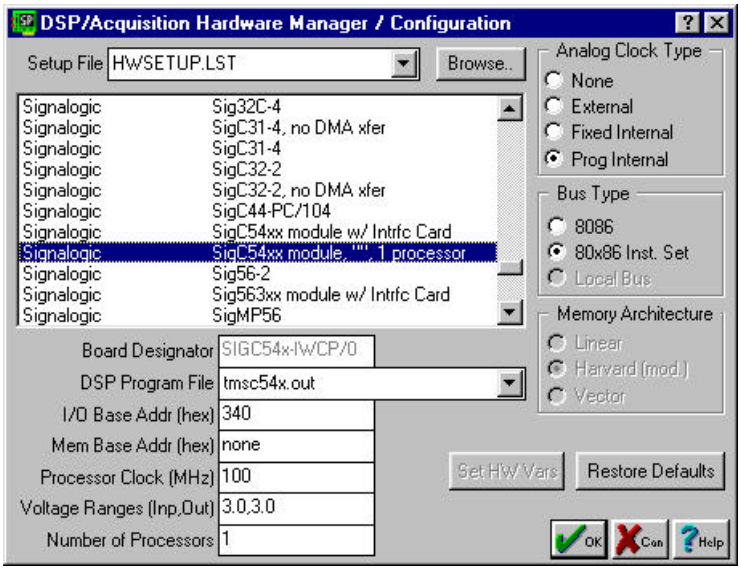

Figure 5-6 Hardware Manager Dialog

11. Click the Download DSP COFF file (top-left) button. Type in the name of the COFF file in the field and Click OK button (as shown in Figure 5-6 below).

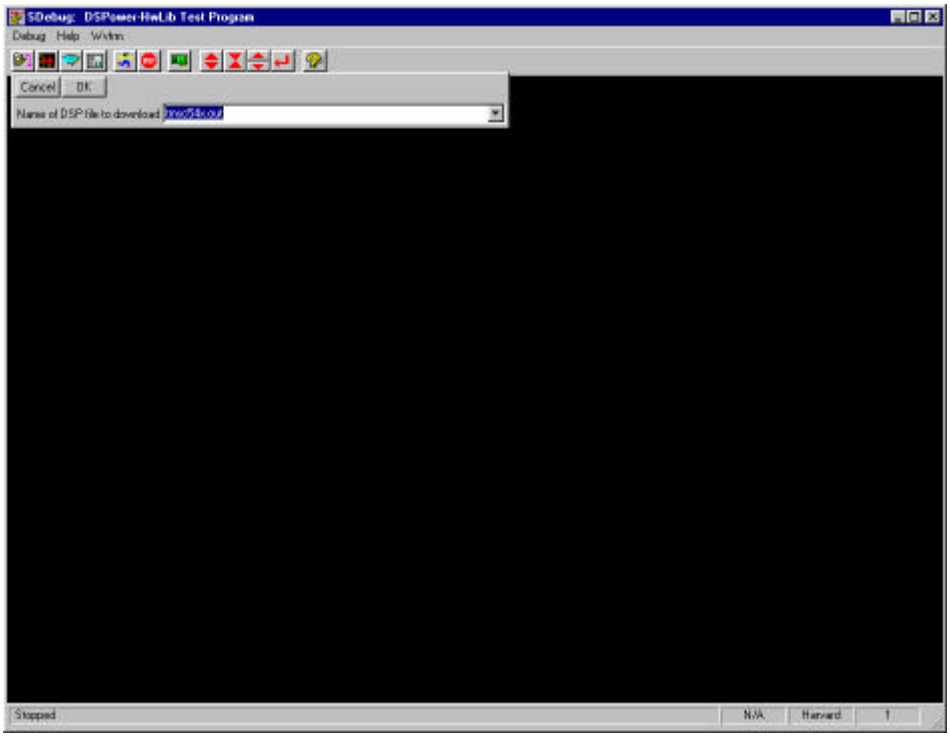

Figure 5-7 DSPower-Debugger Download COFF File

12. Click Display DSP memory Button (the second button from top left). You will see setting for the display as Figure 5-7 below.

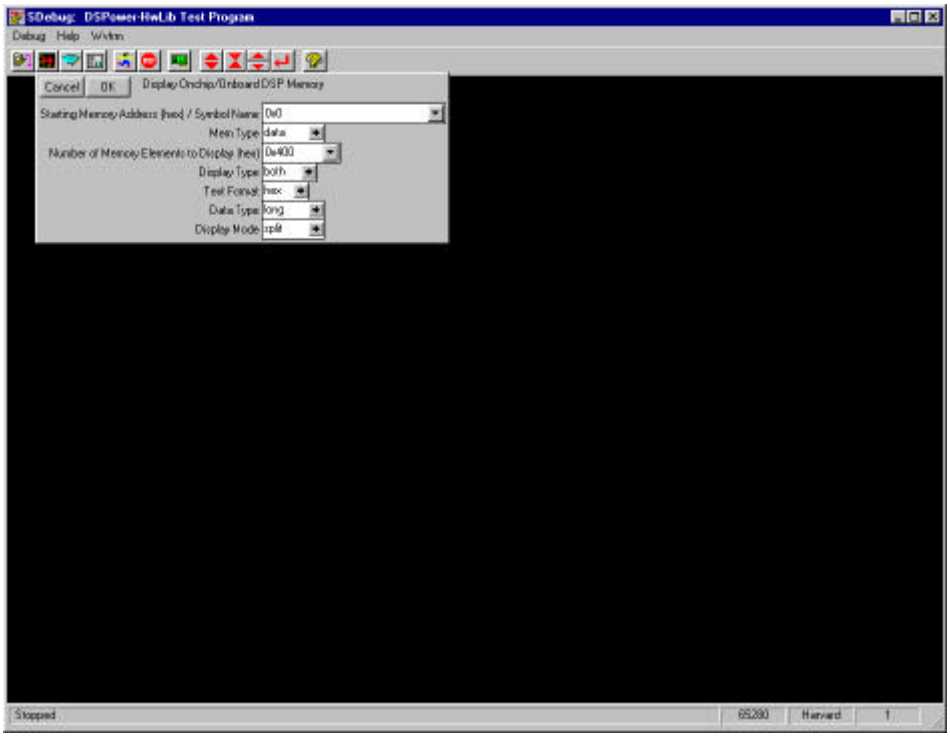

Figure 5-8 Display Onchip/Onboard DSP Memory Settings

13. Select desired settings and Click OK button. You will see some memory displays as in Figure 5-9 below. In this case, the display is in both waveform and text.

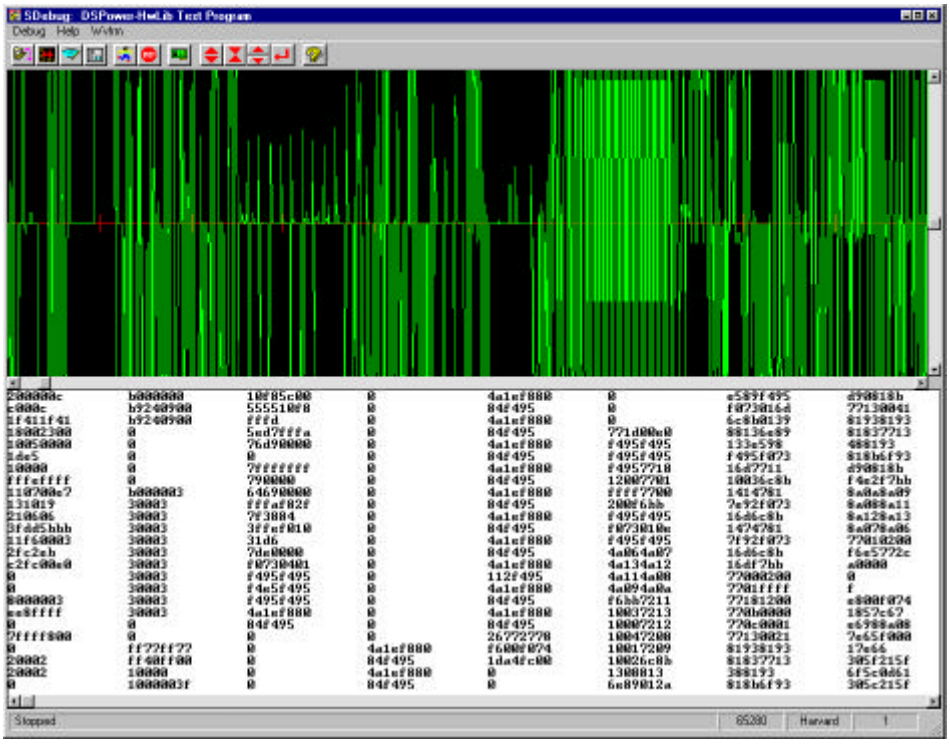

Figure 5-9 DSP Memory Display

5.2 Stand-Alone Mode and UserProc Debugging

In order to build the Source Code Interface in stand-alone mode, you need to load the project and do following:

1. Project → Configurations. You will see the following dialog box as in Figure 5-5 above.
2. Select Standalone and click Set Active button.
3. Click Done.
4. Project → Build or F7 to build the project.
5. The out file for standalone is built and ready to be downloaded through JTAG for debugging.

6.0 Code Composer Studio Debugging Examples

Code Composer Studio offers many helpful debugging tools to assist users and developers.

NOTE: In order to make any kind of accesses to the DSP, you must be using some form of driver that allows CCS to access the hardware. Examples of such connections would be a JTAG emulator, or a parallel port cable. The following techniques will NOT execute correctly without a connection to the DSP through a CCS driver!

Viewing Memory

To view sections of memory on the DSP, choose "View->Memory" from the menu bar in Code Composer Studio.

FIGURE 6-1 HERE

Displaying a Graph of Memory

To view memory location values in the form of a graph, choose "View->Graph" from the menu bar. You will be presented with several different graph style options. Most will choose "Time/Frequency". Choose your preferences from the dialog that pops up:

FIGURE 6-2 HERE

Setting Breakpoints

If you are debugging within Code Composer Studio, it is possible to set, remove, and disable breakpoints in your code. There are two ways to manipulate breakpoints. The first method involves choosing "Debug->Breakpoints" from the menu. The following dialog box will appear:

FIGURE 6-3 HERE

To set breakpoints easily, double-click the desired line of source code. A red dot will appear on the bar to the left of the line, signifying that breakpoint has been created at that line. To remove this breakpoint, double-click the line once more. To disable the breakpoint, single-click the red dot.

7.0 DirectDSP Development Examples

DirectDSP (DSPower-HwLib) software provides direct interface to all supported DSP/data acq. hardware from MATLAB v4 and v5, Borland C/C++ 4.5x and 5.0x, MSVC C/C++ 4.x and 5.x, LabView 4.x, and Visual Basic v4/v5. All interfaces include source code examples and demos, and project files where applicable. For the MATLAB interface, we use a "toolbox" approach similar to other MathWorks toolboxes, except for general DSP/data acq hardware use.

DirectDSP software is complementary to traditional JTAG-based debug and IDE software. DirectDSP provides "bulk transfer" debug capability and "real-time view" into the processor and onboard resources via a fast, non-JTAG method -- for example, host-port, PCI bus, shared memory, etc, depending on the hardware type. One distinct advantage of using DirectDSP to access the board and processor vs. a JTAG based package is that JTAG normally requires the processor to be stopped to data transfers. In contrast, DirectDSP allows the processor to continue to run, including all analog I/O or other activities. For example, from Visual C/C++, Visual Basic, or MATLAB programs, it is possible you can read data buffers and plot them graphically, update parameters in the processor's code or memory data, access variables and buffers by symbol names -- all while the processor continues to run in real-time.

5.1 Visual Studio Development Examples

1. Run Visual C++. Typically, Start → Programs → Microsoft Visual Studio XX → Microsoft Visual C++ XX.

2. Load project dscope32ms.dsw or dscope32ms.dsw located at \dspower\hwlib\ into the workspace.

3. Build the project by Build → Build Dscope32ms.exe or F7. If build successfully, will will see the outputs as in Figure 7-2 below.

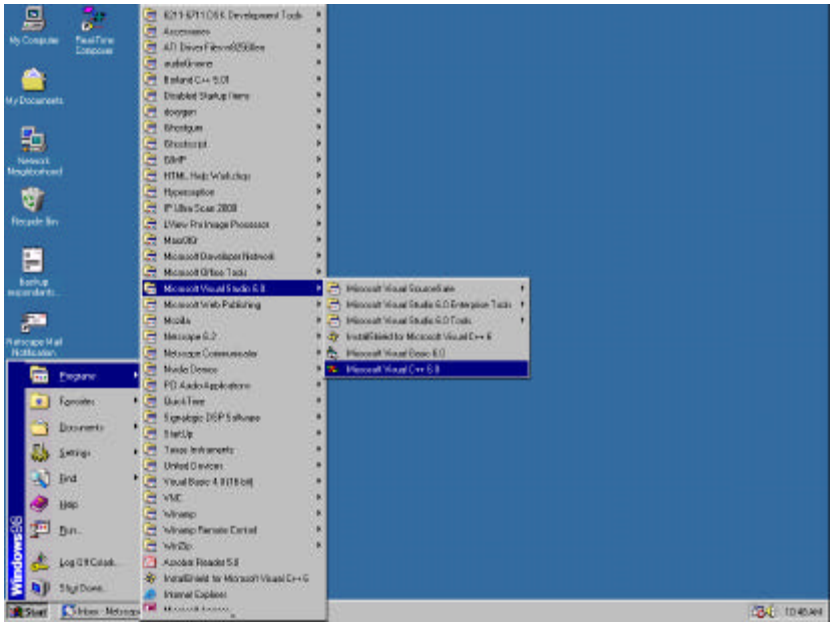

Figure 7-1 Open up Visual C++

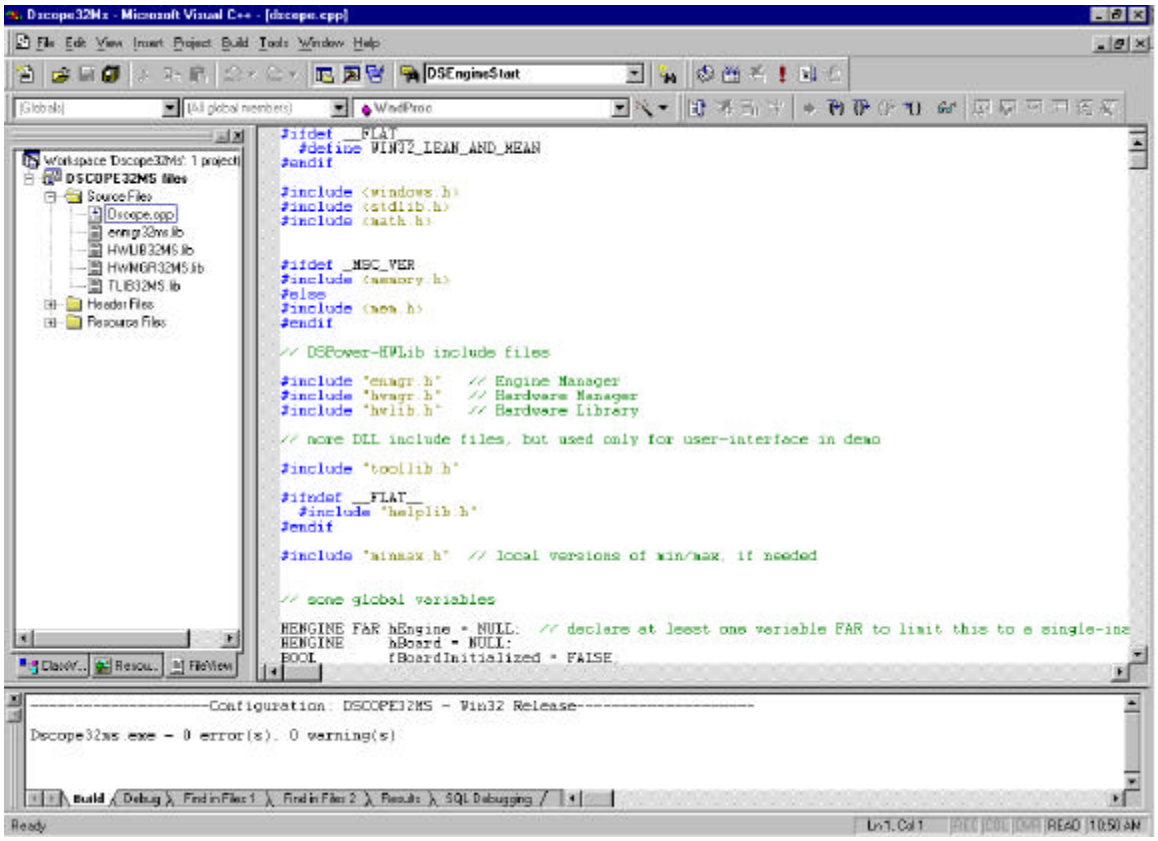

Figure 7-2 Build Dscope Demo Successfully

4. Run the program by Build → Execute Dscope32ms.exe or Ctrl+F5.

5. You will see the Hardware Manager dialog box as in Figure 7-3 below.

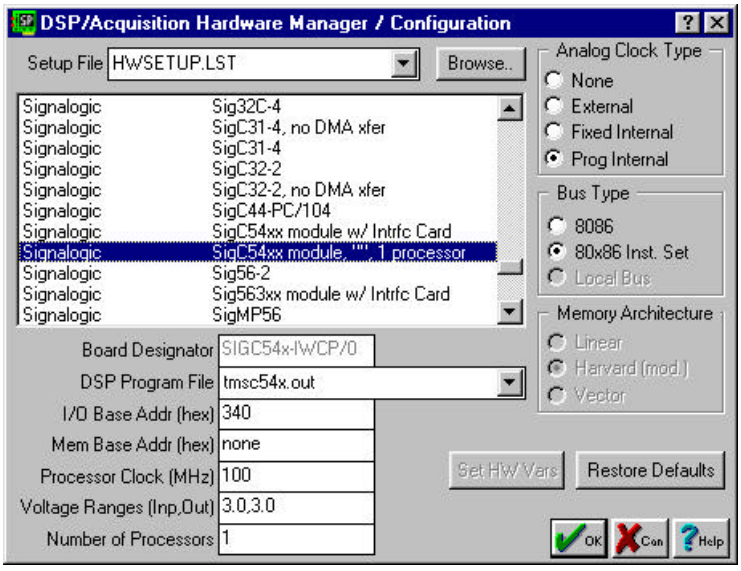

Figure 7-3 Hardware Manager Dialog

6. Choose the proper board type and click Ok, you will see the Digital Scope program running and display some trace. If you don't have any input hooked up to the board, you will just see a flat line.

Figure 7-4 Digital Scope Demo

7. If you click on the left-most button to expand the vertical scale, you will see some noise in the display as in Figure 7-4 below.

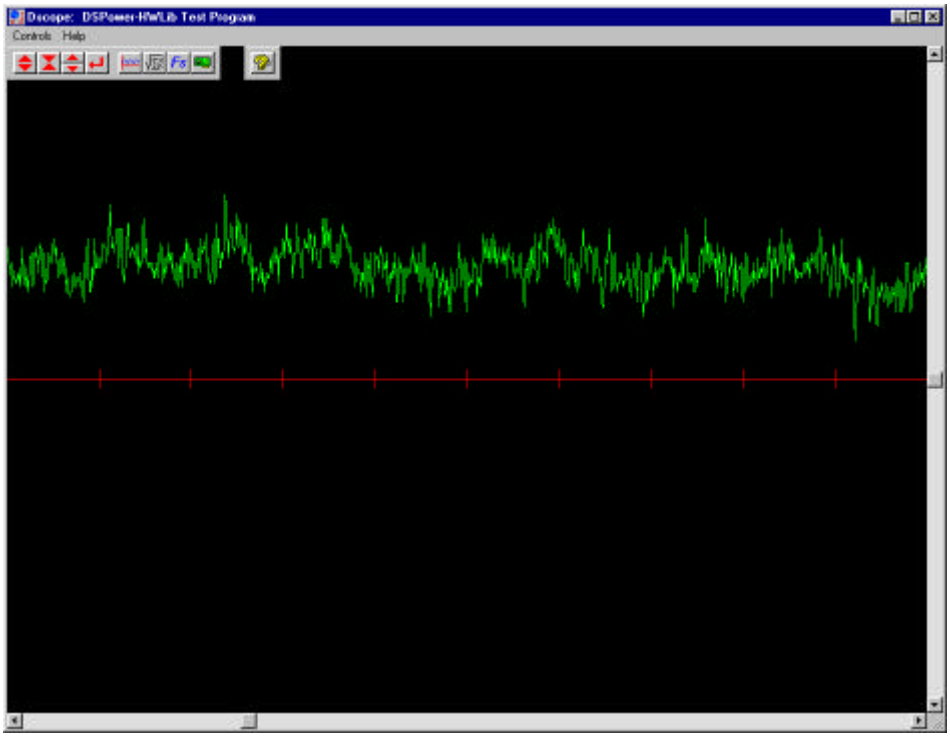

Figure 7-5 Digital Scope with Vertical Scale Expanded to See the Noise

5.2 MATLAB Development Examples

1. Run MATLAB, any version 5.0 or higher.

2. When the Command Window appears, type in proper directory for the Matlab examples as in figure 8-1 below. Matlab Command Window

Figure 7-6 Matlab Command Window

3. For example: Running the hwtest2.m demo program as shown in Figure 7-6 above will bring up a Hardware Manager dialog as in Figure 7-1 above.

4. Choose the proper board type and click Ok.

5. This demo acquires one frame of analog input data from DSP board and displays it to Matlab figure window. If you don't have any input hooked up such as function generator, you will see some noise as in Figure 7-7 below.

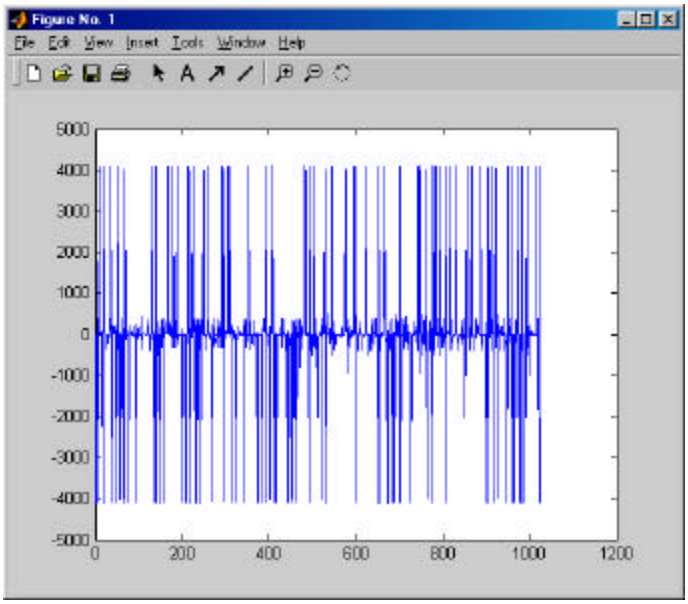

Figure 7-7 MATLAB Figure Window

8.0 Hypersignal Software Example

1. Run Hypersignal-Macro by Start → Programs → Signalogic DSP Software → Hypersignal – Macro.

2. Go to System Config Menu as in Figure 8-1 below. Enter tic54xx-iwcp for C54xx processors and tic6xxx-iwcp for C6xxx processors.

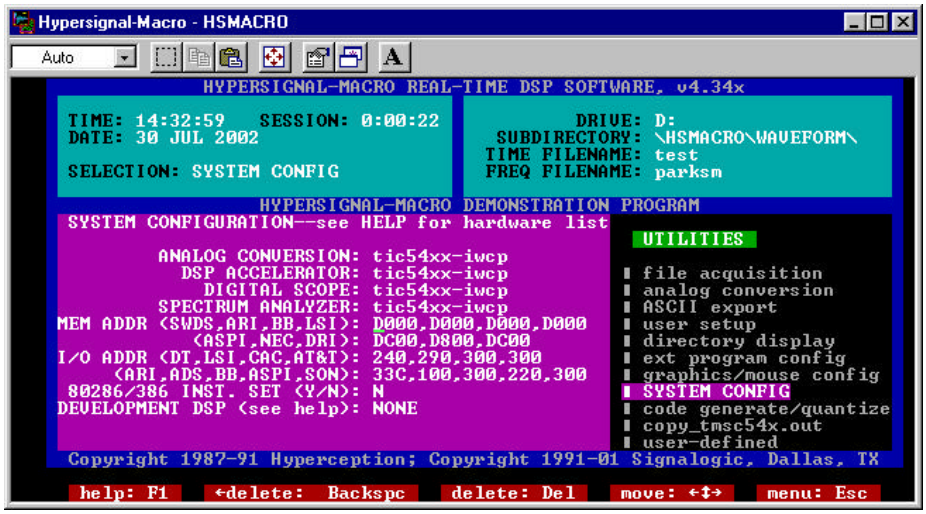

Figure 8-1 Hypersinal-Macro System Configuration

3. Hit ESC key, go to under Time Domain Menu than go to FFT Generation and ENTER.

4. Types in a wave file name such parksm for example as Figure 8-2 below.

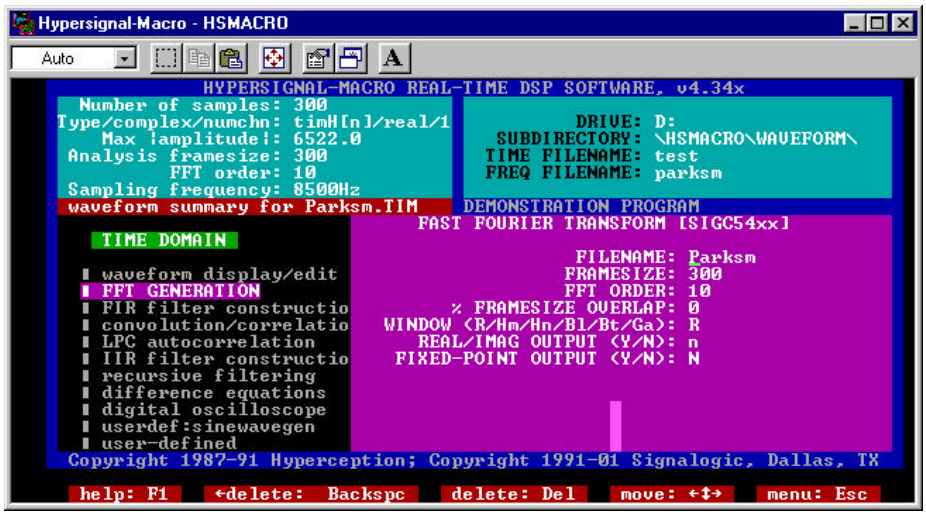

Figure 8-2 Hiypersignal-Macro FFT Generation

5. Hit Ctrl+Enter to run the FFT.

6. If you see a error message comes say invalid interface card address as Figure 8-3 below, you should exit out of Hypersignal-Macro by hitting the following key sequence: ESC, q(quit), a(abort)/e(exit and save).

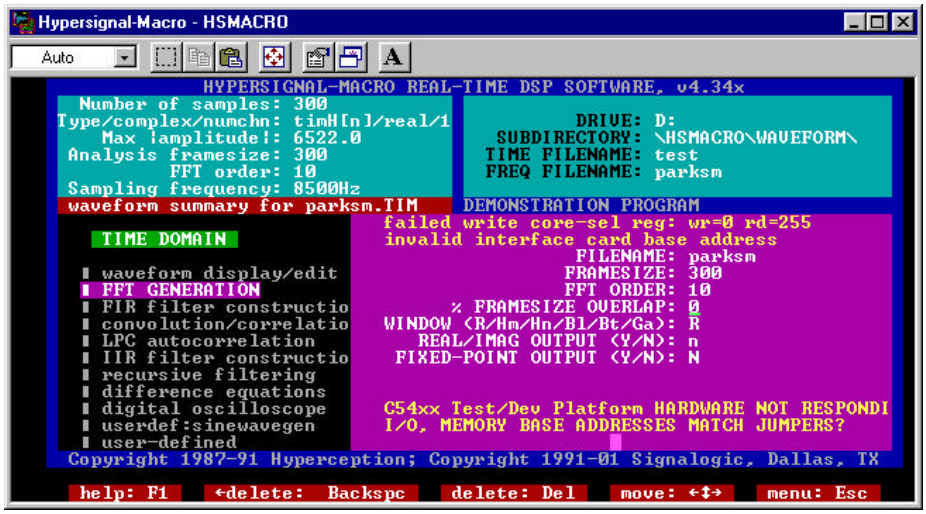

Figure 8-3 Hypersignal-Macro Error Message

7. Wrong setting in the Hypersignal Configuration for the base address could cause this error. To fix that, do the following:

    a. Open hsmacro.cnf in \hsmacro directory in Notepad.

    b. Go to the 5th number from the bottom of the file.

    c. If your have PC/104 card, your base-address should be 0x320; therefore the 5th number from the bottom should be 800.

    d. If your board base address set to 0x340, the 5th number from the bottom should be 832.

    e. Modify the number accordingly, save and exit the file.

    f. Go back to step 7.

8. If you run FFT and see following message as in Figure 8-4 below, press y key.

9. Go to Magnitude Display under Frequency Domain Menu as in Figure 8-5 below.

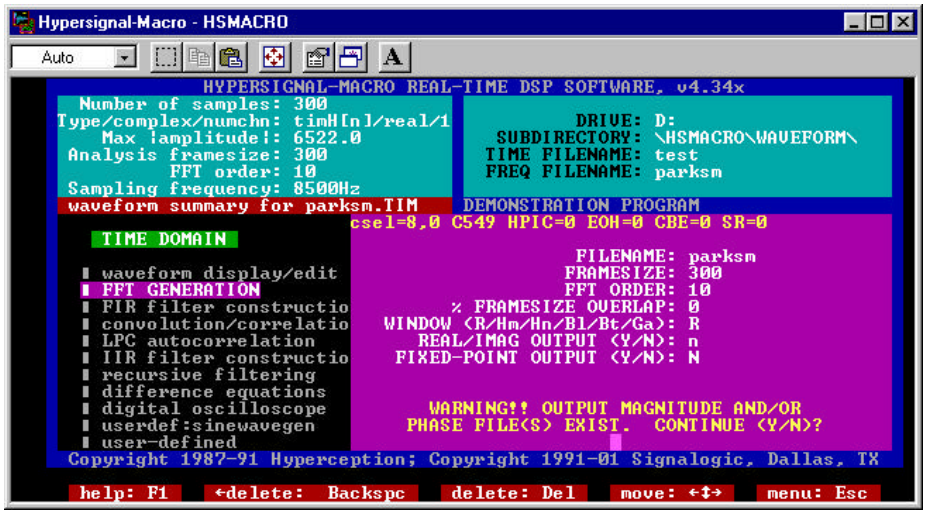

Figure 8-4 Run Hypersignal-Macro FFT Generation

10. Hit Ctrl+Enter

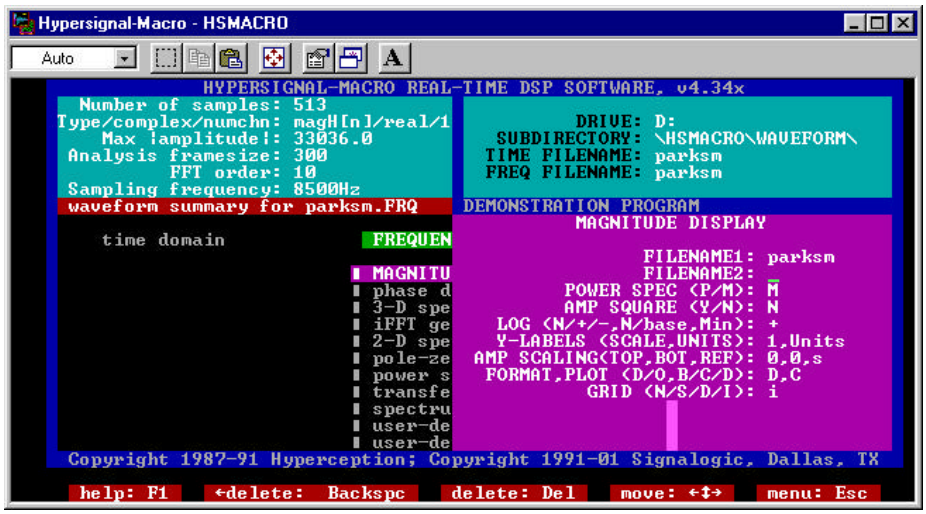

Figure 8-5 Hypersignal-Macro Magnitude Display

11. You see the answer displayed as in Figure 8-6 below.

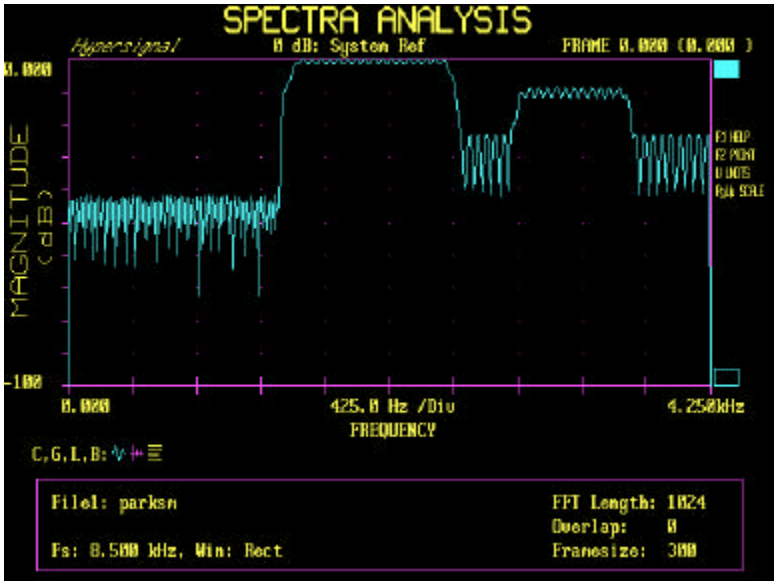

Figure 8-6 Magnitude Display for FFT Generation of Parksm waveform